User's Manual

Autohydro

FOR WINDOWS™

Release 5.0

a u t o s h i p
Systems Corporation

Suite 312
611 Alexander Street
Vancouver • B C
V6A 1E1 • Canada

Tel • (604) 254-4171
Fax • (604) 254-5171
Internet • http://www.autoship.com

Autoship Systems Corporation
Suite 312
611 Alexander Street
Vancouver, B.C.
V6A 1E1
Canada

Tel: (604) 254-4171
Fax: (604) 254-5171
Internet: http//www.autoship.com

Table of Contents

Chapter 1: Getting Started 1

1.0 Introduction.. 1
1.0.1 What Auto*hydro* can do for you.. 2
1.0.2 Auto*ship* Program Suite 3
1.0.2.1 Auto*hydro*.. 4
1.0.2.2 Auto*power*.. 4
1.0.2.3 Auto*build* 4
1.0.2.4 Auto*plate* 4
1.1 System Requirements... 5
1.2 Program Installation.. 6
1.3 Hardware Lock Installation.. 7
1.3.1 General.. 7
1.3.1.1 Description.. 7
1.3.1.2 Precautions.. 7
1.3.2 Windows 95, Local Station.. 8
1.3.3 Windows NT, Local Station.. 9
1.3.4 Hardware Lock Installation, Network System.............................. 10
1.4 Starting and Exiting Autoship... 11
1.5 Updating Auto*ship* 11

Chapter 2: About The Program 13

2.0 Introduction.. 13
2.1 How Autohydro Works... 14
2.1.1 Parts of the Program... 14
2.1.2 Model*maker* 14
2.1.3 Auto*hydro* 15
2.1.4 Commands... 16

2.2 Key concepts and terms 17
2.2.1 File Types 17
2.2.2 Geometry Files 17
2.2.3 Data Hierarchy 18
2.2.3.1 Part 19
2.2.3.2 Component 19
2.2.3.3 Shape 20
2.2.3.4 Stations and Points 20
2.2.3.5 Co-ordinate system 20
2.2.3.6 Units 21
2.2.3.7 Using files from GHS/BHS 21
2.2.4 Typical job flow 22

Chapter 3: A Quick Sailthrough of Autohydro 23

3.0 Introduction 23
3.1 Model*maker* 24
3.1.1 Start Model*maker* 25
3.1.2 Create a part 25
3.1.3 Create component 1 25
3.1.4 Shape component 1 27
3.1.5 Create shaped component 2 28
3.1.6 Join components 29
3.1.7 Save the file 29
3.2 Auto*hydro* 30
3.2.1. Start Auto*hydro* and find lightship displacement 30
3.2.2 Add deck cargo and find resultant drafts 32
3.2.3 Generate and print report 33

Chapter 4-Tutorials 35

4.0 Introduction 35
4.1 Tutorial 1: Model*maker* 36
4.1.1 Add tanks to the barge 36
4.1.2 Create sounding tubes 43
4.1.3 Create a deckhouse 44

4.2 Tutorial 2: Running Auto*hydro* using menus and screens........ 45
4.2.1 Find hydrostatics, cross-curves and hull data.................. 45
4.2.2 Find drafts for a given weight and center of gravity. 46
4.3 Tutorial 3: Model*maker* project using command files............... 48
4.3.1 Create a barge, LOLLIPOP.GF1.............................. 48
4.4 Tutorial 4: Running Auto*hydro* using commands.................... 51
4.4.1 Obtain basic output from LOLLIPOP.GF1........................ 51
4.5 Tutorial 5: Modelling a vessel using Model*maker* commands.. 52
4.5.1 Load a command file.............................. 52
4.5.2 Load a second command file.............................. 53
4.6 Tutorial 6: How to Produce a Stability Book in Auto*hydro*........ 54
4.6.1 Load a RUN file.............................. 54
4.7 Tutorial 7: How to assess damage stability in Auto*hydro* 56

Chapter 1
Getting Started

1.0 Introduction

Use this manual with your computer on and Auto*hydro* running.

Later in Chapter 1 we will tell you how to install and run Auto*hydro*, but first let's look at how to use this manual:

To make the best use of this manual, first read the introduction to Auto*hydro* carefully and get a sense of what Auto*hydro* can do for you and how it works. Then read the short topic on the basic concepts and terms that you need to understand to work effectively with Auto*hydro*. Next, take the guided tour, "A Quick Sailthrough". Then, you can learn and practice other features of the program by running through the tutorials given here. You will find it best to follow the sequence given and not jump ahead or do the tutorials at random because we are trying to progress from simple and easy tasks to increasingly complex ones.

Auto*hydro* uses some key terms and concepts. There is a short section explaining these terms just before the "Quick Sailthrough."

In addition to this manual, you will find further information in the Help system provided with the program. For detailed technical information on any features of the program please see the Auto*hydro* Reference Manual.

1.0.1 What Auto*hydro* can do for you

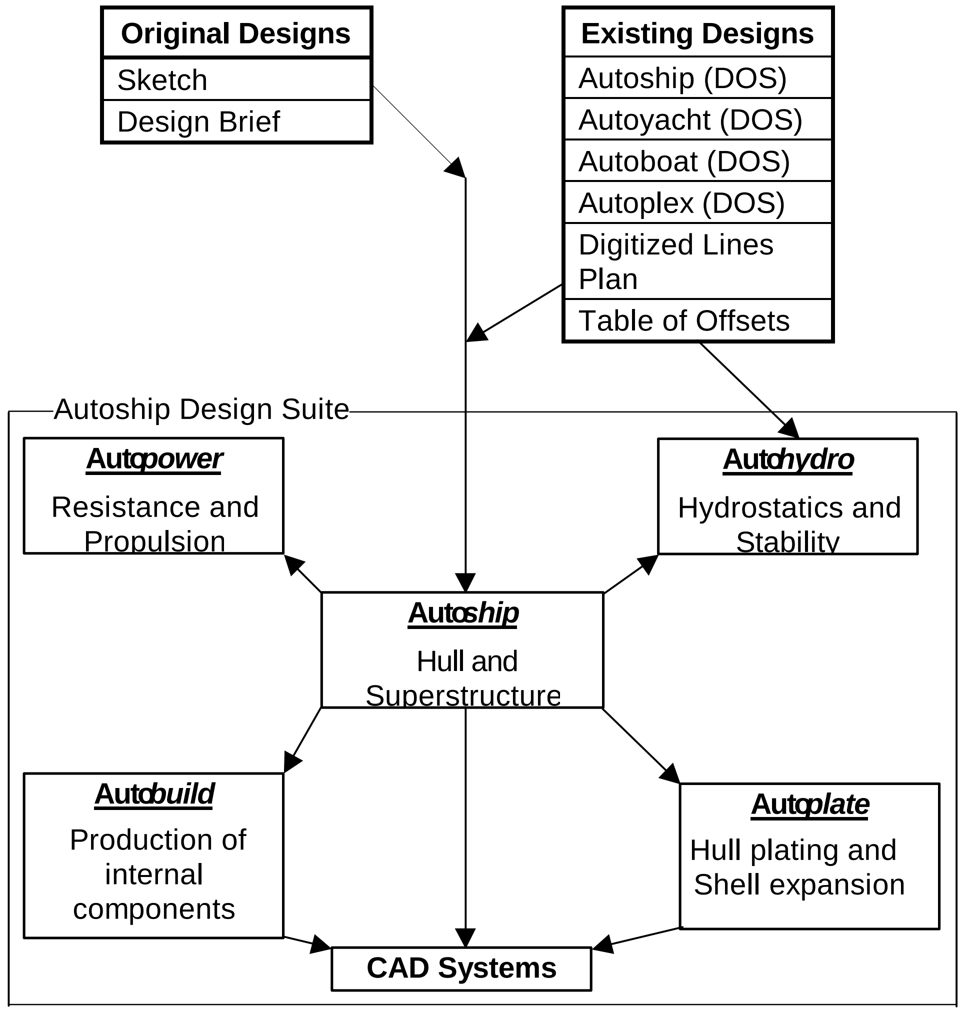
Auto*hydro* has the capability to analyse the hydrostatic characteristics of any kind of vessel under a wide variety of conditions. It is comprised of two main modules:

Model*maker*, is used to model any vessel, from kayaks to supertankers, and also buoys, docks, storage tanks, drilling platforms and anything else that displaces and/or contains fluids.

Auto*hydro*, is used to perform hydrostatic and stability calculations, analyse hydrostatic and stability characteristics for various loading conditions including damage, produce graphic and text for reports such as stability books and tank sounding tables.

1.0.2 Auto*ship* Program Suite

Auto*ship* is part of our integrated suite of Windows programs.



1.0.2.1 Auto*hydro*

Use Auto*hydro* to perform hydrostatic and stability calculations for various loading conditions including damage, and produce graphic and text for reports such as stability books and tank sounding tables. If Auto*hydro* is installed, you can access it from Auto*ship* by selecting Aux-Auto*hydro* from the Auto*ship* menu bar.

1.0.2.2 Auto*power*

Use Auto*power* to perform resistance and power prediction calculations. If Auto*power* is installed, you can access it from Auto*ship* by selecting Aux-Auto*power* from the Auto*ship* menu bar.

1.0.2.3 Auto*build*

Use Auto*build* to design and model internal structures.

1.2.0.4 Auto*plate*

Use Auto*plate* to perform plate expansions for the vessel you designed in Auto*ship*.

Contact us or any of our international dealers for more information on any of our programs.

1.1 System Requirements

The following table shows the minimum and recommended system requirements for running Auto*ship*.

	Minimum	Recommended
CPU	486 processor	Pentium processor
Memory	16 Mb (Windows 95) 24 Mb (Windows NT)	32 Mb (Windows 95) 48 Mb (Windows NT)
Free Hard Disk Space	10 Mb	
Graphics	800x600 resolution, 256 colors *	1024x768 resolution, 32k colors
Mouse	Any Windows pointing device	

*****Operation is possible at 256 colors, but for rendering to function correctly 32k colors or more are needed.

1.2 Program Installation

1. First, you need to install the hardware lock onto your system.

Note: If you do not install the hardware lock, the program will run in test drive (or "demo") mode only, and you will not be able to save, export, or print any of your material or results. (See "Installing the Hardware Lock" below for hardware lock installation details.)

2. Then, with Windows running:

- If you are installing from a CD, run **Setup.exe** found on the CD.
    - If you are installing from floppy disks, run **Setup.exe** found on the Authorization disk.
    - If you are installing from files on your hard drive, run **Setup.exe** on your hard drive.

3. Follow the instructions given by the install program.
4. When you get the message that installation is complete, respond by clicking **OK**.
5. After the installation is complete, you will see these Auto*hydro* icons on the screen: Auto*hydro*, Model*maker* and Print Parts
6. To start Auto*hydro* or Model*maker*, click on the appropriate icon or option from Start – Programs – Autoship.

1.3 Hardware Lock Installation

1.3.1 General

1.3.1.1 Description

A hardware lock is a small electronic device that is installed onto the parallel port of your computer. (Note: If you have a network hardware lock, only one lock is used for all computers on the network. See below for details.) You need the hardware lock in order for Auto*hydro* or Model*maker* to run in normal operating mode. Otherwise, the program will only run in demo mode. Keep the lock in a safe place when not in use. As outlined in the Licensing Agreement, you are exclusively responsible for the hardware lock. You may be required to purchase an additional software license if the hardware lock is lost or stolen.

1.3.1.2 Precautions

- Before beginning the installation, close all running programs.
- Only network locks may be daisy-chained. For a local station installation, a single lock may be configured for all the programs the user is authorized to use.
- Certain local printers may interfere with the operation of the lock. If this is the case, try a different printer driver, or install another parallel port.

1.3.2 Windows 95, Local Station

1. Plug the hardware lock into the parallel port.
2. If you had a printer connected to the same parallel port, plug the printer into the other end of the lock.
3. Install Auto*hydro* as discussed above.

1.3.3 Windows NT, Local Station

1. If the Auto*hydro* installation program detects a Windows NT operating system, you will be prompted: “Would you like to copy the NT lock driver files to your computer?” You must respond “Yes”, unless you have previously installed the lock driver. (The NT lock driver is required so your operating system can communicate with the parallel port.) You will then be asked where you want to copy the NT lock driver files to.
2. The Auto*hydro* installation program will indicate a Readme.wri file which contains instructions on how to run the NT lock driver installation program and will then terminate to allow you to run the lock driver installation.
3. Follow the instructions contained in the Readme.wri file to install the NT lock driver.
4. Re-start the Auto*hydro* installation. This time, when the installation program asks if you would like the NT lock driver files copied to your computer, respond “No”.
5. Follow the instructions to complete the installation of Auto*hydro* onto your computer.

1.3.4 Hardware Lock Installation, Network System

If you have purchased a network lock, you will have received a separate floppy named “Network Lock Driver”.

1. Connect the network lock onto the parallel port on any one of the computers in your network.
2. The Network Lock Driver floppy contains 3 directories:
   - \DOS (for Windows 3.11)
   - \NW (for NetWare)
   - \Win32 (for Windows 95 and Windows NT)

   Each directory contains a readme.txt file. Follow the directions appropriate for your operating system.
3. Install Auto*hydro* onto each station on your network from which it will be run, following the installation instructions above.

1.4 Starting and Exiting Autoship

To start Auto*hydro* go to Start – Programs – Autoship and click on the Auto*hydro* shortcut.

To start Model*maker* go to Start – Programs – Autoship and click on the Model*maker* shortcut.

To start Print Parts go to Start – Programs – Autoship and click on the Print Parts shortcut.

The fastest way to get going with Auto*hydro* is to follow our "Quick Sailthrough" (chapter 3, this manual), and then to work your way through the tutorials starting in chapter 4 of this manual.

For advanced and complex projects, you should take an Auto*hydro* training course. Contact Autoship Systems, or your Autoship dealer for schedules and locations of upcoming courses, or for customised training.

1.5 Updating Autoship

When you update Auto*hydro*, the installation procedure is the same as for a new installation. All necessary program files are updated. No data you have saved will be adversely affected by the update.

Chapter 2
About the Program

2.0 Introduction

Chapter 2 introduces the Auto*hydro* and Model*maker* modules. It discusses how the programs work and how to operate them. Since key concepts and terminology are discussed, it is important that new users read this section.

2.1 How Autohydro Works

2.1.1 Parts of the Program

The Autohydro program is composed of three modules:

- Model*maker*
- Auto*hydro*
- GF Print

This manual focuses on the first two modules, Model*maker* and Auto*hydro.* Model*maker* is used to create and edit the vessel models. Auto*hydro* uses the model produced in Modelmaker to calculate the hydrostatic characteristics of the vessel. Each program module has its own sets screens, functions and commands.

A vessel model, or Geometry File (GF) is merely a collection of groups of 2D cross sections, and their associated attributes, which define the entire model. Each group or “part” describes a particular piece of the vessel model, such as the hull or a compartment.

In both Model*maker* and Auto*hydro* you can work using pull-down menus and/or commands. Simple modelling and calculations by using the menus is demonstrated in the guided tour ("A Quick Sailthrough") provided in this manual while the full set of menus and commands is described in the Reference Manual.

2.1.2 Model*maker*

Model*maker* is used to create and edit the vessel model. The main screen displays the model graphically and also shows the co-ordinates of the vertices that make up the model. Various tools allow you to create and edit the parts and components which define the model.

A model is created by defining parts and their constituent components. The part takes on attributes such as a name, side factor (Center, Port or Starboard), class (Displacer, Container or Sail), contents and specific gravity. Any number of parts can be created, but one of them must be named *Hull*. Without a part called *Hull* Auto*hydro* cannot process the GF. Next, for each part, you define the physical shape, or volume, of the part by creating a component, or group of components, and assign further attributes, such as side designation and permeability. Model*maker* supplies several given basic forms, e.g., Box, Cylinder, etc. for which you define the extents to build your model. The shape of the component may be modified graphically, or by editing it's actual co-ordinates. One component may be joined to another component within the part to form a single, complex volume. You can also fit, or trim, components with each other. By repeating this process of creating, shaping, joining and fitting you end up with a complete model of the vessel.

2.1.3 Auto*hydro*

Using the model created in Model*maker*, Autohydro calculates various hydrostatic values for given conditions. You can analyse the model in three distinct ways, or modes:

Given	**You can find**
a. draft, trim, heel	weight, center of gravity
b. weight(s)	draft, trim, heel
c. waterlines	vessel characteristics

Auto*hydro* shows the vessel's current situation graphically in three different views on the screen, and also as text and values in the "Hydrostatics" window.

Putting together the appropriate sets of instructions allows you to analyse the model in several different conditions.

Various reports describing the characteristics or stability of the vessel are available by using the appropriate pull down menu or command. By assembling the reports in the right order, you can produce a stability book.

2.1.4 Commands

Both Model*maker* and Auto*hydro* can be controlled either by using pull down menus or by commands (instructions) issued by the user.

In Model*maker*, commands are grouped together using the program's text editor and saved as a "COMMAND" (.CMD) file. The command file is then run as a batch file and used to the create vessel model. Using commands is a convenient technique for modelling vessels because you can easily change the configuration of the model or correct mistakes just by editing the command file and re-running it instead of re-creating the model detail by detail as in the interactive mode. Often the best approach to create a model uses a mixture of the two methods.

You can also use a command file to repeat tasks, for example, when you have to model similar tanks. One tank (or any other part) definition can be copied and pasted to create others.

In Auto*hydro*, you can either group commands as a RUN file using the program's text editor, or type individual commands on the command line on the main screen. When run, the RUN file simply instructs Auto*hydro* to perform a certain set of calculations upon the currently loaded vessel model. Often a mixture of single commands and running a .RUN file is the best approach to accomplish a given task.

2.2 Key concepts and terms

2.2.1 File Types

In addition to Geometry files, each program uses other special file formats:

Model*maker*

- .CMD (Command): used to create and edit the vessel model.

Auto*hydro*

- .CD (Condition): information for a vessel loading condition.
- .CRI (Criterion): stability criteria to be used for stability assessment.
- .LIB (Library): of user-defined macro commands.
- .RUN (Run): a set of commands that Auto*hydro* uses to set conditions and perform calculations.
- .SAV (Save): geometry file and information to reconstruct the loading, criteria, and settings currently in effect.

2.2.2 Geometry Files

A GF (Geometry File) contains all the information about the geometry of the vessel, represented in detail appropriate for hydrostatic calculations. It describes the volumes of the hull and various compartments, their permeabily and the contents and specific gravity they hold, or displace. Each volume to be modeled consists of a series of transverse sections, arranged in order from bow to stern. By adjusting the permeability, deductions, such as for bow thrusters can be made. Additionally, volumes which are not considered watertight, but are to be used to for wind calculations can be defined.

Model*maker* and Auto*hydro* will read any GF, including those produced Auto*ship* as well as those created by GHS and BHS.

2.2.3 Data Hierarchy

The GF uses a hierarchical data structure which is broken down into:

i. Parts
ii. Components
iii. Shapes

Each part is made up of one or more components and each component is made up of a shape. For instance, the hull normally consists of one part having several components, such as the hull, keel, appendages and superstructure,

This is how theGFs data structure is arranged:

Data element	Information contained
Part	*A NAME TAG FOR AN ENTITY*
	name (includes side designator, .P, .S or .C)
	class (Displacer, Container, Sail)
	co-ordinates of the “reference point”
	containers only: substance (fluid contents and specific gravity) sounding tube definition
	references one or more components
Component	the volume information
	name (may include side designator)
	permeability factor (negative = deducting)
	shift vector
	references a shape
Shape	the geometrical description
	name
	x, y, z offsets of the component

There are two special considerations regarding GFs:

1. Every model has to have a part called HULL.
2. Every component must be attached to a part.

2.2.3.1 Part

Each part is identified by a name that is unique within the model. The name also has a side designator: .C for centerline, .P for port, or .S for starboard. Each part is assigned a Contents which has a name and a specific gravity, i.e., Salt Water, 1.025. A part has a class designation: Displacer, Container or Sail. Displacers displace the fluid “contents”, Containers contain the fluid “contents”, a Sail part only produces an area for Wind heeling moments - it does not displace or contain a fluid. The part also has a Reference Point - a location that may be used as a reference when loading the tank, or as the spilling point when the tank type is set to Spilling. Tanks may also have a sounding tube definition. Note that a part is only a name tag with attributes and it must reference one or more components to comprise a volume.

While Model*maker* can deal with parts, components and shapes, Auto*hydro* can only access the model at the part level. Therefore, it is important that you define all the volumes you wish to work with, and to group them in a logical fashion.

2.2.3.2 Component

Each component is identified by a name and side designator that must be unique within the referencing part. Each component has an “effectiveness” - i.e. permeability. By default, displacers are assigned 1, containers are assigned .985. Components that are deducted from other parts/components have a negative effectiveness. The component refers to one shape by name and gives the shape a specific location on the vessel by means of the shift vector, which by default is 0,0,0.

2.2.3.3 Shape

The shape is identified by a name, which is unique within the model. It represents a volume in space by a series of 2-dimensional cross sections. Each section is defined by a longitudinal position and a series of transverse and vertical co-ordinates. The shape and has a definite size and orientation but its location is subject to modification by the referencing component.

Note: Normally, you do not have to deal with shapes because they are treated by the program as the characteristics of a component.

2.2.3.4 Stations and Points

You will see each component on the screen as a series of 2-dimensional transverse sections. Each station has points located on its perimeter. You can change the shape of a station by modifying the positions of the points.

2.2.3.5 Co-ordinate system

A right-hand co-ordinate system is used. It is aligned with the principal axes of the vessel as follows:

Origin: The point at which all three of the model axes cross, (the co-ordinates = 0, 0, 0). It can be located at any position along the center plane of the vessel, such as the AP at Baseline, FP at waterline or midships at underside of keel. The first piece of information given to describe the model ultimately establishes the origin, however the origin can be moved at any later time.

L-axis: The longitudinal, or X axis runs along the length of the vessel. Positive is aft of the origin, negative is forward. The program also recognises A for aft and F for forward. The model may be positioned anywhere longitudinally.

T-axis: The transverse, or Y axis runs across the vessel. The transverse origin is the centerline of the vessel. It is positive to starboard and negative to port. The program also recognises S for starboard and P for port. Any Centerline (.C) components will be automatically mirrored across the centerline.

Z-axis: The vertical, or Z axis is perpendicular to the vessel's baseplane (Z=0). Positive is above the baseplane and negative is below. The model may be positioned anywhere vertically.

2.2.3.6 Units

Lengths must be supplied either in decimal feet or meters. Switching from one set of units to another automatically converts the values in the model.

The standard units of measurement used in Model*maker* and Auto*hydro* are:

Measurement	*Metric units*	*Imperial units*
Length	m	ft
Weight/Displacement	MT	LT
Volume	m^3	ft^3

Weight and displacement can also be expressed in Kilograms (KG), Short Tons (ST) and Kilopounds (KP). For more information, see the UNITS command in the Reference manual.

2.2.3.7 Using files from GHS/BHS

Most files from GHS or BHS (from Creative Systems Inc.) may be used interchangeably with Auto*hydro* and Model*maker*:

- GHS GF's can be read directly by Auto*hydro* or Model*maker*.
- GHS Condition files, Criterion files, and Macro files can usually be read directly by Auto*hydro*.
- GHS .RUN files can be read directly by Auto*hydro*.
- Partmaker .RUN files can be read directly by Model*maker* if you rename .RUN to .CMD.

Auto*hydro* and Model*maker* cannot read GHS .SAV files or any other files that represent the environment settings.

Some GHS and Partmaker commands are not essential to Auto*hydro* or Model*maker* but do not affect operation. These non-essential commands are ignored.

2.2.4 Typical job flow

An Auto*hydro* project usually proceeds through three steps:

1. Create a hull using one, or a combination, of these methods:
   - Model*maker* menus
   - a Model*maker* command file
   - digitising offsets in Model*maker*
   - an Auto*ship* GF
2. Create internals such as tanks and sounding tubes using Model*maker*.
3. Create appendages such as derricks etc. using Model*maker*.
4. Perform analyses of the model's hydrostatic characteristics using Auto*hydro*.

Chapter 3
A Quick Sailthrough of Autohydro

3.0 Introduction

This chapter presents a quick tour of Auto*hydro*. This is what we will do:

Objectives

i) To use Model*maker* to build a simple barge (40m x 10m x 5m) with forward and aft rakes (.5m headlog at fwd and aft ends, sloping to the bottom 5m from each end).

ii) To use Auto*hydro* to obtain lightship weight from given drafts and the operating draft for a load condition with deck cargo.

3.1 Model*maker*

Preview of tasks

Model*maker*

3.1.1 Start Model*maker*
3.1.2 Create a hull part
3.1.3 Create component 1
3.1.4 Shape component 1
3.1.5 Create shaped component 2
3.1.6 Join components
3.1.7 Save the file

Auto*hydro*

3.2.1 Start Auto*hydro* and find lightship displacement
3.2.2 Add deck cargo and find resultant drafts
3.2.3 Generate and print a report

*Get ready**!*** Our co-ordinates will originate from:

Longitudinal "0" at midships

Transverse "0" at center line

Vertical "0" at the lowest point on the flat bottom midships.

Remember the data hierarchy: you will make and edit model elements in the following order:

Part

Component

Shape

3.1.1 Start Model*maker*

Click on the Model*maker* icon in the Autoship group/folder. When Model*maker* finishes starting, pick File - New GF. In the dialog that appears, type a file name, MYBARGE.GF1 for instance, specify a directory to store it in, then click on OK.

Note: Make a note of the filename and the directory where you are storing the file.

3.1.2 Create a part

On the top menu bar, click on Options and select Meters.

From the Edit menu select the Part Create/Edit function. In the dialog that appears, under Part Name you will see NEWPART highlighted. Type HULL over it (it does not matter whether you type in upper or lower case).

Note: You must have a part named HULL. If you don't, the model won't work in Auto*hydro*.

For Side, click on the Center radio button. For Class, click on the Displacer radio button. Click on OK. Now, back at the main screen, nothing shows because there are no components to display yet.

3.1.3 Create component 1

We will create the hull as a combination of two box-shaped components. The first component will be the parallel midship section including the aft rake, and we will shape it to form the aft rake. The second component will be the forward rake and, to demonstrate different techniques, we will shape it using a different method. We will then join them together to make a single component.

From the Edit menu, select the Component Create function. In the dialog that appears, you will see HULL on the left hand side. If it is not already highlighted, then click on it to highlight it. On the right hand side you will see a list of the basic component shapes, called "primitives", that Model*maker* uses as building blocks.

Click on the Box radio button and then click on Input.

The Create Box dialog will appear. Under Component Name you will see C1.

Click on the FWD End input box and type 15f (it will show as 15.000f).

Click on the AFT End input box and type 20a

Click on the Station Spacing input box and type 2.5 to specify the space between adjacent stations.

Note: Later, if you find this station spacing too large for fitting parts and components together smoothly, you can change thespacing by adding more stations with the Fill command in the Edit menu.

To the right of OUTBOARD, in the TRANS input box, type 5 to set the half-breadth.

To the right of TOP, in the VERT input box, type 5 to set the height of the deck above baseline.

In the Side Factor box click on Center.

Click on OK.

In the Create Component dialog, click on Done to return to the main screen.

A rectangle will now appear on the screen.

Choose Iso (Isometric) from the View menu. This will display the first component of the model showing all the stations, with the currently selected station (the forward-most one) highlighted in black and the red cross-shaped cursor highlighting the first vertex of the station.

3.1.4 Shape component 1

We will shape the first component in this way:

Shift through the group of stations from the forward-most station to the aft-most by pressing F6 until the aft-most station is highlighted. “Location” near the bottom left of the screen will read 20a.

The cursor will show at the bottom-center of the station highlighting the first vertex of the station. We will change the shape of this section by moving the two bottom vertices up to from the lower edge of the headlog. To do this, press the up arrow on the keyboard repeatedly until the 1st vertical cell under Edit Points at the left of the screen reads 4.5.

Shift the cursor to the next vertex on the station by pressing F4.

Move the cursor up with the up arrow until the 2nd vertical cell under Edit Points reads 4.5. Hit Enter to update the changes and redraw the view.

Shift to the next station forward by pressing F5.

Shape this station differently (just for practice): type 2.25 in the 1st vertical cell and 2.25 in the 2nd vertical cell and hit Enter to update the changes and redraw the view.

Note: If at any time the screen does not show changes that you have made, click the Redraw button.

Now we will create the second component.

3.1.5 Create shaped component 2

Model*maker* allows you to specify a sloped top, bottom, inboard or outboard face for a “box” shape. We will use this feature to create the barges sloped forward rake.

From the Edit menu select the Component Create option. In the Create Component dialog click on HULL, on Box and then on Input.

In the Create Box dialog you will see C2 for the name of this component. Type these values:

FWD End 20f

AFT End 15f

Station Spacing 2.5

Outboard Trans 5

Top Vert 5

Click the SLOPING? checkbox for Bottom. This will open up three additional input boxes. Input the following values:

Vert 4.5

Long 20f

Vert 0

Long 15f

Make sure that Side Factor is set to Center and then click on OK. Click on Done. If the main screen shows no change, select View Part from the View menu to see the entire part.

Note: To scroll through the components of the model, use the space bar.

3.1.6 Join components

We will now join the two components for a cleaner appearance.

Note that whether a part is made of one component or several, there will be no difference in the resultant calculations.

Select View - View Profile to see the entire model in profile.

From the Edit menu choose Join To. The Join dialog will appear. Under Join Component, pick HULL for part and C2 for component.

Under Component To Join To, pick HULL for part and C1 for component.

Click on the External radio button.

Click on OK.

You will see your complete model without any break between components.

Note that all reference to the component C2 is gone - it has been amalgamated with component C1.

3.1.7 Save the file

Choose Save from the File menu.

3.2 Auto*hydro*

3.2.1. Start Auto*hydro* and find lightship displacement

Start Auto*hydro* by clicking on the Auto*hydro* icon in the Autoship group/folder.

When Auto*hydro* has finished starting, click on File - Open, select the GF you just created and click on OK. When the GF is finished loading you should see a representation of your barge.

One of the main ways Autohydro can be used is to have it find the displacement and LCG for a given waterline, and assign that displacement and LCG to the lightship weight. We will input two drafts to define a waterline use this method to determine the lightship weight.

Suppose the lightship drafts are:

1.34m aft (at the forward end of the aft rake, at 15a); and

1.26m fwd (at the aft the of the forward rake, at 15f)

Input these drafts into Auto*hydro* by typing on the command line:

DRAFT 1.34 @ 15a, 1.26 @ 15f <Enter>

Note that the comma is optional and it does not matter whether you type in upper case or lower case.

Now let's get Auto*hydro* to find the displacement and longitudinal center of gravity (LCG) corresponding to these drafts. On the command line, type:

SOLVE WEIGHT LCG <Enter>

In the Hydrostatic Values window you should now see the following results:

> Draft status: Aft = 1.35 Mid = 1.30 Fwd = 1.25 m
>
> Lightship: 419.0 MT
>
> Fixed Weight LCG: .194a

Note that the drafts shown are not the drafts you put in. The drafts reported are being measured at the extreme ends of the vessel, while the drafts you input were located closer to midships.

To reset the locations where the drafts aremeasured, so they match your draft mark locations, typing on the command line:

> DRAFT "US KEEL" 0 @ 15a, 0 @ 15f <Enter>

Now we know the lightship weight and its LCG. By default, both the transverse and vertical center of gravity are zero.

Assuming we had calculated the vertical center of gravity to be 3.2m above baseline, we can input this value with the command:

> VCG 3.2 <Enter>

You may have noticed that when you entered this last command the Solve button, in the top left corner of the screen, changed from grey to red. The solve button is grey red when the vessel in equilibrium and red when the vessel is not in equilibrium. In this case, changing the VCG caused the vessel to go out of equilibrium.

To Auto*hydro*, the vessel is in equilibrium when three conditions are met:

1. the displacement = the total of all weights (lightship + added weights + tank loads)
2. the LCB = trimmed LCG (RAt = 0)
3. the TCB = heeled TCG (RAl = 0)

To get Auto*hydro* to calculate a new equilibrium, click on the Solve button. After a quick calculation, the Solve button will turn grey, the graphics window showing the model will be updated and the Hydrostatics Values window will show new values.

At this point, Auto*hydro* has a model to work with, has a reference line for measuring the drafts and knows the vessel's lightship weight and center of gravity. Now we can add some deck cargo and find the new drafts.

3.2.2 Add deck cargo and find resultant drafts

Auto*hydro* maintains two separate weight categories. One has further sub-categories:

1. Fixed Weight:

    i) Lightship Weight

    can be set:

    - with the WEIGHT command, or
    - by specifying drafts or depth and using the SOLVE WEIGHT command.

    ii) Added Weights

    can be input:

    - with the ADD command, or
    - in a dialog box accessed from the Fixed Weight button.

2. Tank Loads

    can be specified:

    - with the LOAD command, or
    - in a dialog box accessed from the Tank Loads button.

In our barge model we will add two fixed weight items as deck cargo. (If we had defined some tanks in the model, we could have loaded them with some contents, thus adding liquid weight.)

To add a fixed weight item, click on the Fixed Weight button. In the Weight List dialog that comes up, click on Add and change the label New Weight Item#01 to Lumber on deck, set its LCG to 12a, VCG to 6.5, and Weight to 3.

To add the second fixed weight item, click on Add, change New Weight Item#01 to Forklift on deck, set its LCG to 10f, VCG to 6.1, and Weight to 1.3.

Close the dialog box by clicking on OK and return to the main Auto*hydro* screen.

Because some variables have been changed the vessel will be out of equilibrium. However, when you exit the Added Weights dialog, Autohydro will automatically find the new equilibrium.

Now we know the total weight and center of gravity of the vessel and cargo, and the resultant drafts. All we need now is to create a report of this condition and print it out.

3.2.3 Generate and print a report

Auto*hydro* has different commands to produce different outputs. The most commonly used (and most general) is Status, which creates an overall status report of the current loading condition.

To generate a report, on the command line, type: STATUS, then click Enter.

If the Output window is open, it will be filled with an unformatted report.

To see the formatted report, click on the Report button.

You can resize the Report window (like all other windows in Auto*hydro*). It also has scroll bars to move around on the page. Since the Report window is a text editor, you can edit your report right here in Auto*hydro*, or write out a file that can be read directly by Word and then sent to Excel.

To print the report, choose Print from the File menu, or click on the Printer icon.

Replay

You have used Model*maker* to create and edit a vessel model.

You have used Auto*hydro* to calculate hydrostatic behaviour of the model.

You have produced a report of the vessel condition.

Chapter 4
Tutorials

4.0 Introduction

This chapter includes seven tutorials. It is suggested that you work through each tutorial in the sequence presented, as each subsequent tutorial depends upon knowledge gained in the previous one.

The tutorials are:

Tutorial 1: Model*maker* project using menus and screens

Tutorial 2: Running Auto*hydro* using menus and screens

Tutorial 3: Model*maker* project using command files

Tutorial 4: Auto*hydro* using commands

Tutorial 5: Modeling a vessel using Model*maker* commands

Tutorial 6: How to produce a stability book in Auto*hydro*

Tutorial 7: How to assess damage stability in Auto*hydro*

4.1 Tutorial 1: Model*maker* project using menus and screens

The learning objectives here are:

- to create parts by fitting them to other parts.
- to create a tank part with a recessed portion by joining a number of components together.
- to create a tank part with a recessed portion by fitting to it a temporary part and then deleting the temporary part.

4.1.1 Add tanks to the barge created in the Quick Sailthrough.

We will create 6 tanks: Two void tanks: a Forepeak, and an Aftpeak plus four cargo tanks in between, two having a recess on top.

4.1.1.1 Load MYBARGE.GF1 (or the name you used for your geometry file) as created in the Autohydro Sailthrough.

Note: Click on View - ISO to get the best viewing angle.

4.1.1.2 Start by adding more stations to the model so that Model*maker* can fit parts together smoothly.

- From the Edit menu choose Fill
- In the Fill station dialog, in the Maximum Station Spacing field, type .5, and click OK.

This will fill the HULL with more stations and ensure smooth fits.

4.1.1.3 Create the fwd tank:

Part:
(Edit - Part Create/Edit and then Add)

Part name:	FOREPEAK
Side:	Center
Class:	Container
Contents:	SEA WATER

Component:
(Edit - Component Create, select FOREPEAK, click on the Box radio button and then on Input)

FWD:	21f
AFT:	14.5f
Station spacing:	Match Stations to Hull
Outboard Trans:	6
Top Vert:	6
Bottom Vert:	-1

Note: Observe that the dimensions extend beyond the hull. To get Model*maker* to fit, or trim, parts properly, you must specify slightly larger dimensions for the part that is to be fitted to another part.

4.1.1.4 Fit FOREPEAK to HULL:

- From the Edit menu choose Fit to.
- In the Fit Component dialog, under Fit Component, for Part Name choose FOREPEAK and the appropriate component.
- Under Part or Component to Fit To, for Part Name choose HULL and the appropriate component.
- Under Type of Fit choose Internal.
- Click OK.

4.1.1.5 Create the aft tank:

Part:

Part name:	AFTPEAK
Side:	Center
Class:	Container
Contents:	Seawater

Component: (Box)

FWD:	14.5a
AFT:	21a
Station spacing:	Match Stations to Hull
Outboard Trans:	6
Top Vert:	6
Bottom Vert:	-1

4.1.1.6 Fit AFTPEAK to HULL:

- From the Edit menu choose Fit to.
- In the Fit Component dialog, under Fit Component, for Part Name choose AFTPEAK.
- Under Part or Component to Fit To, for Part Name choose HULL.
- Under Type of Fit choose Internal.
- Click OK.

We have now created and fit two simple tanks. The next two tanks are more complicated in shape as they have a well, or recess in the top.

We will construct the next tank out of several “boxes” and then join them together.

4.1.1.7 Create a port tank aft of midships, out of 3 components:

Part:

Part name:	NO1
Side:	Port
Class:	Container
Contents:	Diesel oil

Component: (Box)

FWD:	5a
AFT:	14.5a
Station spacing:	Match Stations to Hull
Outboard Trans:	6
Top Vert:	6
Bottom Vert:	-1

Component, for the *second* of the 3 components: (Box)

FWD:	0
AFT:	5a
Station spacing:	Match Stations to Hull
Inboard Trans:	2
Outboard Trans:	6
Top Vert:	6
Bottom Vert:	-1

Component, for the *third* of the 3 components: (Box)

FWD:	0
AFT:	5a
Station spacing:	Match Stations to Hull
Outboard Trans:	3
Top Vert:	3
Bottom Vert:	-1

Join the third component to the second component:

- Click on Edit - Join To. Under Join Component, for Part Name pick NO1.P, for Comp Name pick C3.
- Under Component to Join to, for Part Name pick NO1.P, for Comp Name pick C2.
- Click on External and then OK.

Join the second component to the first component in the same manner.

Now you have a tank with a recess in the top.

4.1.1.8 Fit NO1.P to HULL:

- From the Edit menu choose Fit to.
- In the Fit Component dialog, under Fit Component, for Part Name choose NO1.P and the appropriate component.
- Under Part or Component to Fit To, for Part Name choose HULL and the appropriate component.
- Under Type of Fit choose Internal.
- Click OK.

4.1.1.9 Create a starboard tank aft of midships, matching the NO1.P tank

We will create this tank in a different way. We will start by creating a regular cubic shape. Next, we will create a temporary smaller cubic shape. Then we will fit the larger one to the smaller one. Finally, we will delete the temporary part.

Part: (the tank part)

Part name:	NO1
Side:	Starboard
Class:	Container
Contents:	Diesel oil

Component: (Box)

FWD:	0
AFT:	14.5a
Station spacing:	Match Stations to Hull
Outboard Trans:	6
Top Vert:	6
Bottom Vert:	-1

Part: (the temporary part)

Part name:	Temp
Side:	Starboard

Component: (Box)

FWD:	0
AFT:	5a
Station spacing:	Match Stations to Hull
Inboard Trans:	-1
Outboard Trans:	2
Top Vert:	7
Bottom Vert:	3

4.1.1.10 Fit NO1.S against the part called Temp:

- From the Edit menu choose Fit to.
- In the Fit Component dialog, under Fit Component, for Part Name choose NO1.S.
- Under Part or Component to Fit To, for Part Name choose TEMP.
- Under Type of Fit choose External.
- Click OK.

4.1.1.11 Fit NO1.S to HULL:

- From the Edit menu choose Fit to.
- In the Fit Component dialog, under Fit Component, for Part Name choose NO1.S
- Under Part or Component to Fit To, for Part Name choose HULL
- Under Type of Fit choose Internal.
- Click OK.

4.1.1.12 Delete the temporary Part:

- Go to Edit - Delete Part/Component, pick TEMP.S and click on the Delete Part/Comps button. Click on 'Yes' at the message "Delete part TEMP.S" and then click on Close.

That finishes the two tanks with the recesses in the top.

4.1.1.13 Create a port tank fwd of midships:

Part:

Part name:	NO2
Side:	Port
Class:	Container
Contents:	Gasoline

Component: (Box)

FWD:	14.5f
AFT:	0
Station spacing:	Match Stations to Hull
Outboard Trans:	6
Top Vert:	6
Bottom Vert:	-1

4.1.1.14 Fit NO2.P to HULL:

- From the Edit menu choose Fit to.
- In the Fit Component dialog, under Fit Component, for Part Name choose NO2.P.
- Under Part or Component to Fit To, for Part Name choose HULL.
- Under Type of Fit choose Internal.
- Click OK.

4.1.1.15 Create the opposite part, NO2.S:

- In the Edit menu, click on Opposite part.
- In the Opposite Part dialog choose NO2.P from the first. The second field will show NO2.S.
- Click OK.

Now your barge is fitted with all six tanks.

4.1.2 Create sounding tubes for the cargo tanks.

4.1.2.1 Set up the sounding tubes:

- In the edit menu, click on Sounding Tube.
- In the Sounding Tube dialog, choose the tank to have the sounding tube; start with NO1.P.
- Click on Insert points twice.
- In the grid, type in the following values for points 1 and 2. Point 1 <u>must</u> be the lowest point on the tube. Be sure to click on Record to register the changes.

	Long	Trans	Vert
Pt. 1	.2a	4.8p	.005
Pt. 1	.2a	4.8p	5.25

4.1.2.2 Create sounding tubes for the other tanks in the same way. *Remember to designate aft or forward, and port or starboard*.

4.1.3 Create a deckhouse.

4.1.3.1 Create a deckhouse with the following dimensions:

length: 5m - fwd end at 14a, aft end at 19a
height: from the deck to 2.5m above deck,
width: 3m wide - inboard at 2.5m from centerline, the outboard side overhangs the starboard side of the hull by .5m.

Part:

Part name:	Deckhouse
Side:	Starboard
Class:	Sail

Component: (Box)

FWD:	14a
AFT:	19a
Station spacing:	Match Stations to Hull
Inboard Trans:	2.5
Outboard Trans:	5.5
Top Vert:	7.5
Bottom Vert:	5

The modifications to the model are now finished. Save your model. To avoid overwriting your original model, use a different name; we suggest incrementing the extension, i.e., if you named the original barge MYBARGE.GF1, name this model MYBARGE.GF2.

4.2 Tutorial 2: Running Auto*hydro* using menus and screens

4.2.1 Find hydrostatics, cross-curves and hull data

4.2.1.1 Load MYBARGE.GF2 (or the name you used for your geometry file).

4.2.1.2 From the Calculate menu, choose Hydrostatics and in the Drafts dialog, enter the following values:

- First .5
- Last 4.5
- Step .5

Auto*hydro* will show the calculation results in the output window. As well, they will be available in the report window.

4.2.1.3 From the Calculate menu, choose Cross curves and in the Drafts dialog, enter the following values:

- First .5
- Last 4.5
- Step .5

Auto*hydro* will show the calculation results in the output window. As well, they will be available in the report window.

4.2.1.4 From the Calculate menu, choose Hull data. In the Hull Characteristics dialog click on Depth, Trim, Heel. For Vessel Attitude, set values for depth, trim and heel.

Auto*hydro* will show the calculation results in the output window. As well, they will be available in the report window.

4.2.2 Find drafts for a given weight and center of gravity.

4.2.2.1 On the Command line, type <weight 100 .25a 0 3.6>. Weight is a command telling Auto*hydro* to set the lightship weight and center to the values that follow; the first value is the weight, the second is the, the third is tcg, and the fourth is the vcg. Note that the lcg, tcg and vcg are optional, but they will remain at the default, 0, until changed by some other method.

WEIGHT

Auto*hydro* maintains two weight categories, for use in calculations:

i. Fixed weight
ii. Liquid weight

Fixed weight

There are two kinds of fixed weight:

i. lightship weight: this is <u>set</u> with the WEIGHT command, or <u>found</u> with the SOLVE WEIGHT… command.
ii. the weight of all other fixed items, as set with the ADD command.

Liquid weight

The volume of contents in a tank is set either with the LOAD command or in the Tank List dialog. You can set the load either by load fraction, or by weight, or, if sounding tubes are defined, by sounding. Setting or changing one of these will update the others in a given tank. You can also change the tank contents by using the CONTENTS command, or by selecting a fluid from the Fluid list in the Tank List dialogue.

4.2.2.2 Load tanks: Click on the tank icon and set the loads of some of the tanks in the Tank List dialogue. Click on OK when you are done.

4.2.2.3 On the command line, type STATUS to generate a report of the current loading scheme and resultant vessel attitude. View the results in the report window.

4.2.2.4 One the command line, type RA to generate a righting arm curve for the current loading scheme. View the results in the report window.

4.3 Tutorial 3: Model*maker* project using command files

The learning objectives here are:

- to use Model*maker*'s text editor to write a .CMD file (a series of commands).
- to run the commands to produce a vessel model.

4.3.1 Create a barge, LOLLIPOP.GF1

We will create a barge which has a deckhouse and two deck-mounted cylindrical diesel oil tanks. The hull dimensions are 40m long, 10m wide, 5m deep, with forward and aft rakes sloping from 15f and 15a, respectively, to .5m below deck.

4.3.1.1 Click on CMD on the menu bar. A text editor will appear.

– If the text editor window is *not empty*, then click on the Files - New menu option to clear the editor.
– Click any where in the editor window and type CLEAR. The CLEAR command removes any existing geometry from Model*maker*'s memory.

Note: Model*maker* cannot create a new part if a part with the same name already exists. Therefore, when re-running a .CMD file it is important to remove the previously created set of parts before creating new ones.

– Below the CLEAR command, type the following, pressing ENTER after every line: (an explanation of each command follows in italics)

units mt	*set units to meters and tonnes*
create hull.c	*start description of part "hull.c"*
spacing .5	*set station spacing at .5m intervals*
ends 15f 15a	*define two ends of a box-like component*
top 5	*define the top of the box*
bottom 0	*define the bottom of the box*
outboard 5	*define the outboard face of the box*
component fwdrake	*describe another component for hull*
ends 20f 15f	*define the two ends of another box*
top 5	*define the top of this box*
bottom 0 @ 15f 4.5 @ 20f	*define a sloping bottom*
outboard 5	*define the outboard face of this box*
join hull.c	*join this component to "hull.c"*
component aftrake	*describe another component of "hull.c"*
ends 15a 20a	*define the top of this box*
top 5	*define the top of this box*
bottom 0 @ 15a 4.5 @ 20a	*define a sloping bottom*
outboard 5	*define the outboard face of this box*
join hull.c	*join this component to hull.c*
/	*end description of part "hull.c"*
create deckhouse.c	*start description of part "deckhouse.c"*
class sail	*set to sail rather than container*
ends 14a 17.5a	*define two ends of a box-like component*
top 8	*define the top of the box*
bottom 5	*define the bottom of the box*
outboard 1.5	*define the outboard face of the box*
/	*end description of part "deckhouse.c"*

create cargo.s	*start description of part "cargo.s"*
contents do	*set the contents to diesel oil*
cylinder 6f 2.4 4.7 7a 2.4 4.7 3.2 3.2	*define as a cylinder*
/	*end description of part "cargo.s"*
create cargo.p	*start description of part "cargo.p"*
contents do	*set the contents to diesel oil*
opposite cargo.s	*copy and mirror shape of "cargo.s"*
/	
write c:\autoship\hydro\samples\lollipop.gf1	
	save the geometry

Check that the path, as specified in the last command, is valid for your computer.

Note: A convenient way to write this series is simply to copy and paste similar instructions: you can copy the six lines for the forward rake (from "component fwdrake" to "join hull") and paste them as the next segment, for the aftrake. *But remember to change "fwd" for "aft" and the co-ordinates from "f" to "a"*.

4.3.1.2 Still in the text editor, click on Run, and choose Restart. The text editor window will close and Model*maker* will begin processing the set of commands.

4.3.1.3 When processing has finished, from the View menu, choose whichever view you prefer and the View all option. You will see your model on the screen. If you don't, click on Redraw in Shape editor box.

You have now finished creating the barge that we will be used for Tutorial 4.

4.4 Tutorial 4: Running Auto*hydro* using commands

The learning objective here is:

- to use Auto*hydro* commands to perform simple calculations.

4.4.1 Obtain basic output from LOLLIPOP.GF1

4.4.1.1 Start Auto*hydro* and click on the Editor icon. In the Text Editor that appears, type the following series of commands:

- read c:\autoship\hydro\samples\lollipop.gf1
- ghs 2 3 4
- cc 2 3 4
- hull /dr 2 3 4 /da

The “read” command must contain the same path as specified in the “write” command from the previous tutorial.

4.4.1.2 Click on RUN and then Batch in the Text Editor top bar. Auto*hydro* will calculate hydrostatic curves, cross curves and the hull characteristics at drafts of 2m, 3m and 4m and display the results in both the output window and the report window.

4.5 Tutorial 5: Modelling a vessel using Model*maker* commands

The learning objectives here are:

- to use Model*maker's* text editor to construct a command sequence
- to understand the operation of the locus command
- to save the command sequence as a .CMD file
- to run the .CMD file

Note: The locus command allows you to define a station by specifying a longitudinal location and a set of transverse-vertical co-ordinate pairs. The locus command is particularly useful when you wish to create a GF from a table of offsets.

TIP An easy way to work with a table of offsets is to put the offsets into a spreadsheet. You can then simply cut and paste them into a command file.

4.5.1 Load a command file

4.5.1.1 Open the Command File editor and load TUT5A.CMD from the …\hydro\sample directory. Take a moment to look at the contents of the command file. You will see several "Locus" commands. In TUT5A.CMD, the exact offsets of a hull have been entered as LOCUS commands.

4.5.1.2 In the top bar of the text editor, click on Run - Restart. This will cause Model*maker* to execute the set of Locus commands and produce a hull.

4.5.2 Load a second command file

4.5.2.1 Load the command file TUT5B.CMD, replacing TUT5A.CMD currently in the text editor. This command file will create all the tanks for the hull just created. Note that the first two commands, Clear and Read, have the character "`" at the front of the line to "comment out" the command (not have it run). Normally, you would start your .CMD files with these two commands, however, because the model from TUT5A.CMD has not been saved, we have commented the commands out here. Note also the use of the Temp(orary) part and the command Delete at the end of the .CMD file to delete it.

4.5.2.2 In the top bar of the text editor, click on Run, and choose Restart to create all the tanks.

Click on View - Iso and then View - View Part and then press repeatedly on the space bar to toggle through all the parts.

4.5.2.3 Save the completed GF file with a name and directory of your choice.

4.6 Tutorial 6: How to Produce a Stability Book in Auto*hydro*

The learning objectives here are:

- to construct a typical sequence of commands for intact stability analysis as follows:
  a) Set up limits
  b) Specify lightship weight (Alternatively, you can specify drafts)
  c) Repeat the following steps for each loading condition
     i. Set up the loading condition
     ii. Solve for equilibrium
     iii. Report the results

4.6.1 Load a RUN file

4.6.1.1 In Auto*hydro,* click on Open and load the GF file you just saved in Model*maker*.

4.6.1.2 Click on the Edit icon to open the run file editor. Load the file TUT6.RUN by clicking on File - Open and selecting the file from the ..\hydro\samples directory.

Take a moment to review the contents of this file. It may be helpful to refer to the Reference Manual, or the On-line Help system, to understand what the various commands and parameters will accomplish and why they are used.

With this run file, we will:

- Set up limits
- Specify l/s weight(s)
- [Alternatively, you could specify drafts.]
- Set up a load condition
- Solve for equilibrium.
- Report the results
- Clear the loads and added weights
- Set up a second loading condition
- Solve for equilibrium
- Report the results

4.6.1.3 Run the RUN file by clicking on Run in the top bar of the editor. Observe that a Stop sign appears in the top bar of Auto*hydro* while the calculations are being carried out. When the top sign disappears, the Report Window will open showing the results. You can either print them out, save them or discard them.

4.7 Tutorial 7: How to assess damage stability in Auto*hydro*

The learning objectives here are:

- to construct a typical sequence of commands for damaged stability analysis

4.7.1.1 To be sure that you will start with the program initialised correctly, click on Open and reload the GF file you had saved in Model*maker*.

4.7.1.2 Click on the Edit icon to start the Run file editor. Load the file TUT7.RUN. With this file we will:

- Define limits
- Set up lightship
- Set up a loading condition
- Set up a damage condition
- Solve for equilibrium
- Report the results
- Clear the loads and added weights
- Set up a second damage condition
- Solve for equilibrium
- Report the results